**Numerical Simulation for Transient Flow of Automobile Engine Turbocharger**

Zhang Qi¹, Zheng Sifa, Han Qiang, Qi Songming, Liu Zhongda

Suzhou Automobile Research Institute, Tsinghua University, Suzhou 215134, China

¹email: johnchee@sina.com

**Key words:** turbo charger; compressor; aeroacoustics; fluid dynamics

**Abstract:** In order to explore the aerodynamic noise mechanism of automobile engine turbocharger compressor, a 3-dimensional unsteady mathematical model was established for turbocharger compressor. Numerical simulation for the transient flow of the turbocharger compressor was carried out. The paper focused on the characteristics of pressure pulsation in the compressor. The simulation results show the process of pressure fluctuation excitation from blade rotation in the compressor, which provides the fluid dynamics information to study the compressor aerodynamic noise mechanism.

**Introduction**

A turbocharger that has been widely used, has important application values in improving the vehicle dynamic performance and energy consumption. However, the car with a turbocharger, will has a poor vibration and noise performance, which affects the ride comfort. The aeroacoustics characteristic of turbocharger compressor is of the most significant, which concerns domestic and aboard scholars[1-3]. Liu lianyun et al.[4] studied on the acoustic performance of mufflers of the compressor turbocharger by CFD and experimental methods. Li huibin et al.[5] made a steady numerical simulation for the turbocharger compressor. The broadband noise model was used to predict the acoustic characteristic of the turbocharger compressor. Even et al.[6] researched the problem of whose noise caused by European diesel engine during the instantaneous acceleration, defined the critical turbulent vibration area on compressor characteristic curve, and proved that the compressor working point too close to the critical surging area is the root cause of the whose noise generated when instantaneous acceleration.

The aerodynamic noise of the turbocharger compressor was basically influenced by fluid dynamics, In this paper, a transient numerical simulation for the fluid dynamic characteristics of the compressor was carried out. The pressure fluctuation properties in the turbocharger compressor were mainly analyzed

**Physical model**

Assumptions for the turbocharger compressor inner flow were made as follow:

1. Fluid in compressor based on the assumption of ideal gas.
2. Using RNG $k - \varepsilon$ turbulence model to consider the effect of turbulence in compressor.
3. Ignore the effect of gravity and friction.

Supported by Natural Science Found of Jiangsu Province(NO. BK20140373)
Mathematical model

According to the above physical model, the mathematical model is established as follows.

(1) Continuity equation

\[ \frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \vec{v}) = S_m \]  

Where \( S_m \) is the source term, and \( S_m \) equals to 0 because of without consider the chemical reaction.

(2) Momentum equation

\[ \frac{\partial}{\partial t} \left( \rho \vec{v} \right) + \nabla \cdot (\rho \vec{v} \vec{v}) = -\nabla p + \nabla \left[ \mu \left( \nabla \vec{v} + \nabla \vec{v}^T \right) \right] + \rho \vec{g} + \vec{F} \]  

Where \( \rho \vec{g} \) is gravity, \( \vec{F} \) is other volume force, they are all equal to 0.

(3) Energy equation

\[ \frac{\partial}{\partial t} (\rho E) + \nabla \cdot (\rho \vec{v} E) = \nabla \cdot (k_{\text{eff}} \nabla T) + S_e \]  

Where \( E = \sum_{i=1}^{\#} \alpha_i \rho_i E_i / \sum_{i=1}^{\#} \alpha_i \rho_i \), \( T = \sum_{i=1}^{\#} \alpha_i \rho_i T_i / \sum_{i=1}^{\#} \alpha_i \rho_i \).

(4) State equation

\[ p = \rho RT \]  

(5) Turbulent equation

Using RNG \( k - \varepsilon \) turbulence model to consider the effect of turbulence in compressor. The model contains the turbulent kinetic energy equation and turbulent dissipation rate equation.

Turbulent kinetic energy equation:

\[ \frac{\partial}{\partial t} (\rho k) + \nabla \cdot (\rho \vec{v}k) = \nabla \cdot \left( \alpha_k \mu_{\text{eff}} \frac{\partial k}{\partial x_i} \right) + G_k + G_b - \rho \varepsilon - Y_m \]  

Turbulent dissipation rate equations:

\[ \frac{\partial}{\partial t} (\rho \varepsilon) + \nabla \cdot (\rho \vec{v} \varepsilon) = \nabla \cdot \left( \alpha_\varepsilon \mu_{\text{eff}} \frac{\partial \varepsilon}{\partial x_i} \right) + \frac{\varepsilon}{k} \left( G_k + C_{\text{vis}} G_b - C_{\text{vis}} \rho \frac{\varepsilon^2}{k} - R_\varepsilon \right) \]  

In these equations, \( G_k \) represents the generation of turbulence kinetic energy due to the mean velocity gradients. \( G_b \) is the generation of turbulence kinetic energy due to buoyancy. \( Y_m \) represents the contribution of the fluctuating dilatation in compressible turbulence to the overall dissipation rate. The quantities \( \alpha_k \) and \( \alpha_\varepsilon \) are the inverse effective Prandtl numbers for \( k \) and \( \varepsilon \), respectively.

(6) Computational domain and boundary conditions

The computational domain is as shown in Figure 1. Point A is at the center of the section away from the exit 30 mm. The pressure of point A variations with time was monitored.

The inlet boundary condition was selected as pressure inlet condition, the inlet pressure is approximately equals to atmospheric pressure:
\[ p_{\text{inlet}} = 0 \]  \hspace{1cm} (7)

The outlet boundary condition was selected as pressure outlet condition:

\[ p = p_{\text{out}} \quad , \quad T = T_0 \]  \hspace{1cm} (8)

Where \( p_{\text{out}}, T_0 \) equal to the atmospheric environmental value.

![Figure 1. Computational domain and mesh.](image)

**Results and Analysis**

The transient CFD simulation was carried out under the condition of inlet and outlet of compressor were both face to atmospheric environmental condition. The transient pressure fluctuation characteristics of the compressor flow field was analyzed. The compressor speed is 10000 rpm.

![Figure 2. The evolution characteristics of the compressor internal pressure pulsation.](image)

Figure 2 shows the evolution of pressure pulsation in the compressor at 0.3-2.8ms. When 0.3ms, the compressor blades initially rotate to compress the air, the pressure distribution of the
flow field is relatively flat. There gradually appeared local high pressure area at the outlet of the compressor blade, when 1ms. The local high pressure propagation in the flow field, and gradually fall off from the outlet part of the compressor blade in the process of compressor blade rotation. The pressure formed fluctuation in the time domain.

Figure 3 shows the pressure of the monitoring point A variation with time. The pressure variation is inconsiderable at the initial stage, and become stronger with the rotation of the turbo blades. The maximum fluctuating pressure difference is 1.27kPa. The pressure fluctuation occurs because blades rotate to compress the air. And this fluctuation will cause blade rotation noise.

![Figure 3. The pressure variation with time at point A.](image)

**Conclusion**

Based on the transient numerical simulation of the turbocharger compressor’s inner flow, some conclusions were obtained as follow:

1. Through the fluid dynamics simulation, the evolution of the pressure pulsation caused by the compressor blade rotation can be obtained.
2. With the compressor blade rotation, there will gradually appear local high pressure area at the outlet part of the blade, and this high pressure area gradually falls off from the outlet of the blade, then propagated in the compressor, with formed fluctuation pressure.

**Reference**


